Lecture notes on

Manufacturing Technology

By

SOMESH SAHU

ASSISTANT PROFESSOR

DEPARTMENT OF MECHANICAL ENGINEERING

HI-TECH INSTITUTE OF TECHNOLOGY

KHURDA,BHUBANESWAR – 752057

CONTENTS

UNIT I

THEORY OF METAL CUTTING

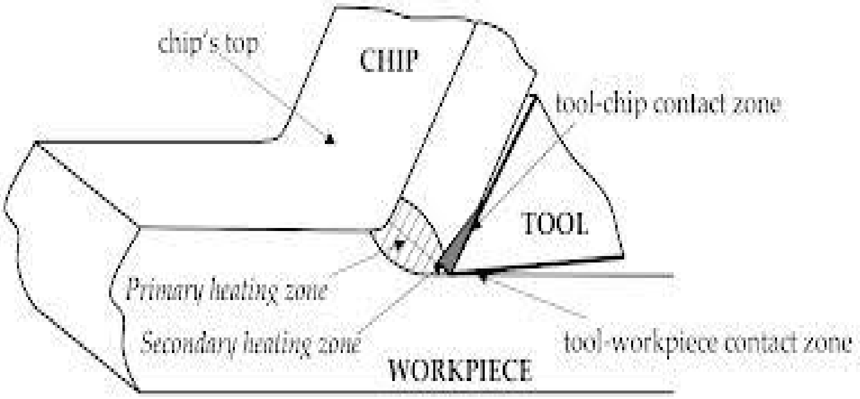
Definitions

Machining: Term applied to all material-removal processes

Metal cutting: The process in which a thin layer of excess metal (chip) is removed by a wedge-shaped single-point or multipoint cutting tool with defined geometry from a work piece, through a process of extensive plastic deformation

1.1 MECHANICS OF CHIP FORMATION

The cutting itself is a process of extensive plastic deformation to form a chip that is removed afterward. The basic mechanism of chip formation is essentially the same for all machining operations. Assuming that the cutting action is continuous, we can develop so-called continuous model of cutting process.



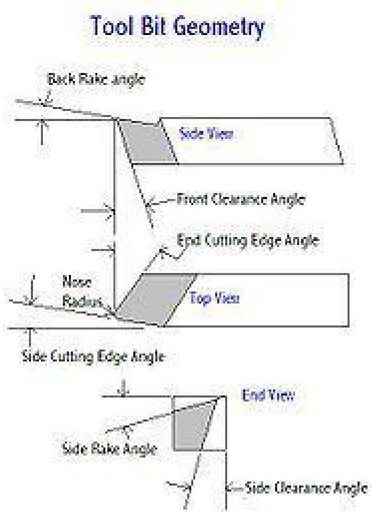
The cutting model shown above is oversimplified. In reality, chip formation occurs not in a plane but in so-called primary and secondary shear zones, the first one between the cut and chip, and the second one along the cutting tool face.

1.2 Single-point cutting tool,

As distinguished from other cutting tools such as a The cutting edge is ground to suit a particular machining operation and may be re sharpened or reshaped as needed. The ground tool bit is held rigidly by a tool holder while it is cutting.

Back Rake is to help control the direction of the chip, which naturally curves into the work due to the difference in length from the outer and inner parts of the cut. It also helps counteract the pressure against the tool from the work by pulling the tool into the work.

Side Rake along with back rake controls the chip flow and partly counteracts the resistance of the work to the movement of the cutter and can be optimized to suit the particular material being cut. Brass for example requires a back and side rake of 0 degrees while aluminum uses a back rake of 35 degrees and a side rake of 15 degrees. Nose Radius makes the finish of the cut smoother as it can overlap the previous cut and eliminate the peaks and valleys that a pointed tool produces. Having a radius also strengthens the tip, a sharp point being quite fragile.



All the other angles are for clearance in order that no part of the tool besides the actual cutting edge can touch the work. The front clearance angle is usually 8 degrees while the side clearance angle is 10-15 degrees and partly depends on the rate of feed expected.

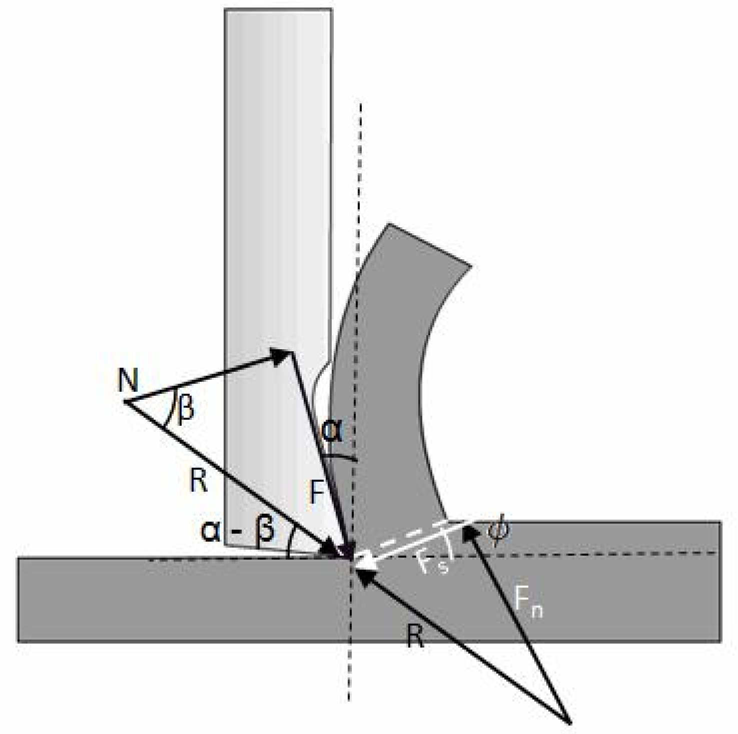
Minimum angles which do the job required are advisable because the tool gets weaker as the edge gets keener due to the lessening support behind the edge and the reduced ability to absorb heat generated by cutting.

The Rake angles on the top of the tool need not be precise in order to cut but to cut efficiently there will be an optimum angle for back and side rake.

1.3 Forces in machining

If you make a free body analysis of the chip, forces acting on the chip would be as follows.

At cutting tool side due to motion of chip against tool there will be a frictional force and a normal force to support that. At material side thickness of the metal increases while it flows from uncut to cut portion. This thickness increase is due to inter planar slip between different metal layers. There should be a shear force (Fs) to support this phenomenon. According to *shear plane theory* this metal layer slip happens at single plane called shear plane. So shear force acts on shear plane. Angle of shear plane can approximately determined using *shear plane theory* analysis. It is as follows $\phi = 45^o + \frac{\alpha}{2} - \frac{\beta}{2}$



Forces acting on the chip on tool side and shear plane side

Shear force on shear plane can be determined using shear strain rate and properties of material. A normal force (Fn) is also present perpendicular to shear plane. The resultant force (R) at cutting tool side and metal side should balance each other in order to make the chip in equilibrium. Direction of resultant force, R is determined as shown in Figure.

1.4 Types of chip

There are three types of chips that are commonly produced in cutting,

Discontinuous chips

Continuous chips

Continuous chips with built up edge

A discontinuous chip comes off as small chunks or particles. When we get this chip it may indicate,

Brittle work material

Small or negative rake angles

Coarse feeds and low speeds

A continuous chip looks like a long ribbon with a smooth shining surface. This chip type may indicate,

Ductile work materials

Large positive rake angles

Fine feeds and high speeds

Continuous chips with a built up edge still look like a long ribbon, but the surface is no longer smooth and shining. Under some circumstances (low cutting speeds of ~0.5 m/s, small or negative rake angles),

Work materials like mild steel, aluminum, cast iron, etc., tend to develop so-called built-up edge, a very hardened layer of work material attached to the tool face, which tends to act as a cutting edge itself replacing the real cutting tool edge. The built-up edge tends to grow until it reaches a critical size (~0.3 mm) and then passes off with the chip, leaving small fragments on the machining surface. Chip will break free and cutting forces are smaller, but the effects is a rough machined surface. The built-up edge disappears at high cutting speeds.

Chip control

Discontinuous chips are generally desired because

They are less dangerous for the operator

Do not cause damage to workpiece surface and machine tool

Can be easily removed from the work zone

Can be easily handled and disposed after machining.

There are three principle methods to produce the favourable discontinuous chip:

Proper selection of cutting conditions

Use of chip breakers

Change in the work material properties

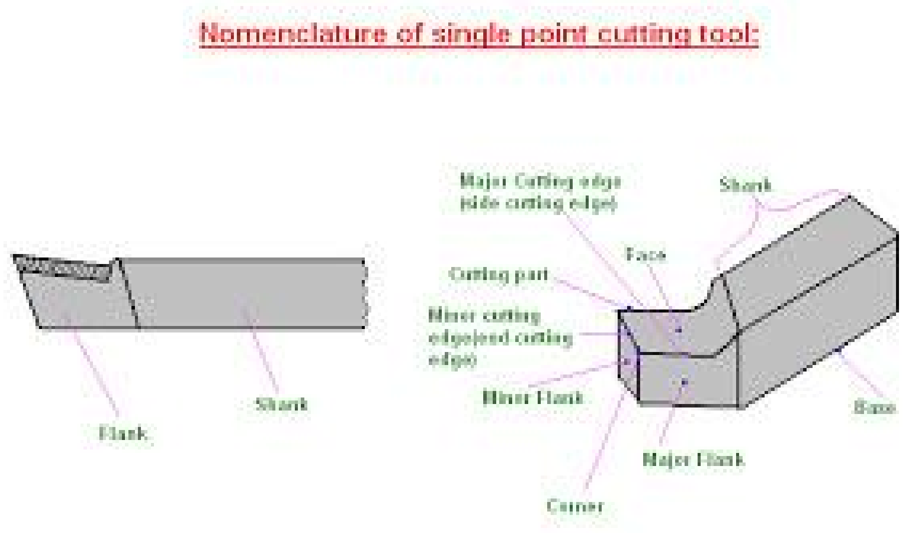
Chip breaker

Chip break and chip curl may be promoted by use of a so-called chip breaker. There are two types of chip breakers

External type, an inclined obstruction clamped to the tool face

Integral type, a groove ground into the tool face or bulges formed onto the tool face

1.5 Cutting tool nomenclature



Back Rake is to help control the direction of the chip, which naturally curves into the work due to the difference in length from the outer and inner parts of the cut. It also helps counteract the pressure against the tool from the work by pulling the tool into the work.

Side Rake along with back rake controls the chip flow and partly counteracts the resistance of the work to the movement of the cutter and can be optimized to suit the particular material being cut. Brass for example requires a back and side rake of 0 degrees while aluminum uses a back rake of 35 degrees and a side rake of 15 degrees.

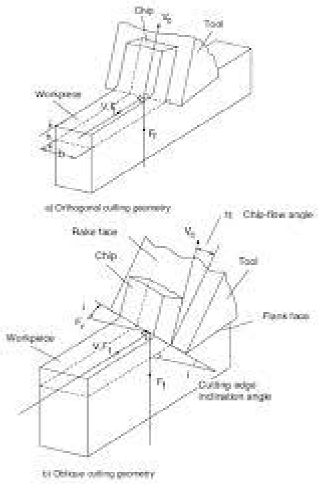
Nose Radius makes the finish of the cut smoother as it can overlap the previous cut and eliminate the peaks and valleys that a pointed tool produces. Having a radius also strengthens the tip, a sharp point being quite fragile.

All the other angles are for clearance in order that no part of the tool besides the actual cutting edge can touch the work. The front clearance angle is usually 8 degrees while the side clearance angle is 10-15 degrees and partly depends on the rate of feed expected.

Minimum angles which do the job required are advisable because the tool gets weaker as the edge gets keener due to the lessening support behind the edge and the reduced ability to absorb heat generated by cutting.

The Rake angles on the top of the tool need not be precise in order to cut but to cut efficiently there will be an optimum angle for back and side rake.

1.6 Orthogonal metal cutting



Orthogonal metal cutting	Oblique metal cutting
Cutting edge of the tool is perpendicular to the direction of tool travel.	The cutting edge is inclined at an angle less than 90° to the direction of tool travel.
The direction of chip flow is perpendicular to the cutting edge.	The chip flows on the tool face making an angle.
The chip coils in a tight flat spiral	The chip flows side ways in a long curl.
For same feed and depth of cut the force which shears the metal acts on smaller areas. So the life of the tool is less.	The cutting force acts on larger area and so tool life is more.
Produces sharp corners.	Produces a chamfer at the end of the cut
Smaller length of cutting edge is in contact with the work.	For the same depth of cut greater length of cutting edge is in contact with the work.
Generally parting off in lathe, broaching and slotting operations are done in this method.	This method of cutting is used in almost all machining operations.

Depending on whether the stress and deformation in cutting occur in a plane (two-dimensional case) or in the space (three-dimensional case), we consider two principle types of cutting:

Orthogonal cutting the cutting edge is straight and is set in a position that is perpendicular to the direction of primary motion. This allows us to deal with stresses and strains that act in a plane.

Oblique cutting the cutting edge is set at an angle.

According to the number of active cutting edges engaged in cutting, we distinguish again two types of cutting:

Single-point cutting the cutting tool has only one major cutting edge

Examples: turning, shaping, boring

Multipoint cutting the cutting tool has more than one major cutting edge

Examples: drilling, milling, broaching, reaming. Abrasive machining is by definition a process of multipoint cutting.

Cutting conditions

Each machining operation is characterized by cutting conditions, which comprises a set of three elements:

Cutting velocity: The traveling velocity of the tool relative to the work piece. It is measured in m/s or m/min.

Depth of cut: The axial projection of the length of the active cutting tool edge, measured in mm. In orthogonal cutting it is equal to the actual width of cut.

Feed: The relative movement of the tool in order to process the entire surface of the work piece. In orthogonal cutting it is equal to the thickness of cut and is measured in mm.

1.7 Thermal aspects

In cutting, nearly all of energy dissipated in plastic deformation is converted into heat that in turn raises the temperature in the cutting zone. Since the heat generation is closely related to the plastic deformation and friction, we can specify three main sources of heat when cutting,

Plastic deformation by shearing in the primary shear zone

Plastic deformation by shearing and friction on the cutting face

Friction between chip and tool on the tool flank

Heat is mostly dissipated by,

The discarded chip carries away about 60~80% of the total heat

The workpiece acts as a heat sink drawing away 10~20% heat

The cutting tool will also draw away ~10% heat

If coolant is used in cutting, the heat drawn away by the chip can be as big as 90% of the total heat dissipated. Knowledge of the cutting temperature is important because it:

Affects the wear of the cutting tool. Cutting temperature is the primary factor affecting the cutting tool wear can induce thermal damage to the machined surface. High surface temperatures promote the process of oxidation of the machined surface. The oxidation layer has worse mechanical properties than the base material, which may result in shorter service life. Causes dimensional errors in the machined surface. The cutting tool elongates as a result of the increased temperature, and the position of the cutting tool edge shifts toward the machined surface, resulting in a dimensional error of about 0.01~0.02 mm. Since the processes of thermal generation, dissipation, and solid body thermal deformation are all transient, some time is required to achieve a steady-state condition

Cutting temperature determination

Cutting temperature is either measured in the real machining process, or predicted in the machining process design. The mean temperature along the tool face is measured directly by means of different thermocouple techniques, or indirectly by measuring the infrared radiation, or examination of change in the tool material microstructure or micro hardness induced by temperature. Some recent indirect methods are based on the examination of the temper color of a chip, and on the use of thermo sensitive paints.

There are no simple reliable methods of measuring the temperature field. Therefore, predictive approaches must be relied on to obtain the mean cutting temperature and temperature field in the chip, tool and work piece.

For cutting temperature prediction, several approaches are used:

Analytical methods: there are several analytical methods to predict the mean temperature. The interested readers are encouraged to read more specific texts, which present in detail these methods. Due to the complex nature of the metal cutting process, the analytical methods are typically restricted to the case of orthogonal cutting.

Numerical methods: These methods are usually based on the finite element modeling of metal cutting. The numerical methods, even though more complex than the analytical approaches, allow for prediction not only of the mean cutting temperature along the tool face but also the temperature field in orthogonal and oblique cutting.

1.8 Cutting tool materials

Requirements

The cutting tool materials must possess a number of important properties to avoid excessive wear, fracture failure and high temperatures in cutting, the following characteristics are essential for cutting materials to withstand the heavy conditions of the cutting process and to produce high quality and economical parts:

Hardness at elevated temperatures (so-called hot hardness) so that hardness and strength of the tool edge are maintained in high cutting temperatures:

Toughness: ability of the material to absorb energy without failing. Cutting if often accompanied by impact forces especially if cutting is interrupted, and cutting tool may fail very soon if it is not strong enough.

Wear resistance: although there is a strong correlation between hot hardness and wear resistance, later depends on more than just hot hardness. Other important characteristics include surface finish on the tool, chemical inertness of the tool material with respect to the work material, and thermal conductivity of the tool material, which affects the maximum value of the cutting temperature at tool-chip interface.

Cutting tool materials

Carbon Steels

It is the oldest of tool material. The carbon content is 0.6~1.5% with small quantities of silicon, Chromium, manganese, and vanadium to refine grain size. Maximum hardness is about HRC 62. This material has low wear resistance and low hot hardness. The use of these materials now is very limited.

High-speed steel (HSS)

First produced in 1900s. They are highly alloyed with vanadium, cobalt, molybdenum, tungsten and Chromium added to increase hot hardness and wear resistance. Can be hardened to various depths by appropriate heat treating up to cold hardness in the range of HRC 63-65. The cobalt component give the material a hot hardness value much greater than carbon steels. The high toughness and good wear resistance make HSS suitable for all type of cutting tools with complex shapes for relatively low to medium cutting speeds. The most widely used tool material today for taps, drills, reamers, gear tools, end cutters, slitting, broaches, etc.

Cemented Carbides

Introduced in the 1930s. These are the most important tool materials today because of their high hot hardness and wear resistance. The main disadvantage of cemented carbides is their low toughness. These materials are produced by powder metallurgy methods, sintering grains of tungsten carbide (WC) in a cobalt (Co) matrix (it provides toughness). There may be other carbides in the mixture, such as titanium carbide (TiC) and/or tantalum carbide (TaC) in addition to WC.

Ceramics

Ceramic materials are composed primarily of fine-grained, high-purity aluminum oxide (Al2O3), pressed and sintered with no binder. Two types are available:

White, or cold-pressed ceramics, which consists of only Al2O3 cold pressed into inserts and sintered at high temperature.

Black, or hot-pressed ceramics, commonly known as cermets (from ceramics and metal). This material consists of 70% Al2O3 and 30% TiC. Both materials have very high wear resistance but low toughness; therefore they are suitable only for continuous operations such as finishing turning of cast iron and steel at very high speeds. There is no occurrence of built-up edge, and coolants are not required.

Cubic boron nitride (CBN) and synthetic diamonds

Diamond is the hardest substance ever known of all materials. It is used as a coating material in its polycrystalline form, or as a single-crystal diamond tool for special applications, such as mirror finishing of non-ferrous materials. Next to diamond, CBN is the hardest tool material. CBN is used mainly as coating material because it is very brittle. In spite of diamond, CBN is suitable for cutting ferrous materials.

1.9 Tool wear and tool life

The life of a cutting tool can be terminated by a number of means, although they fall broadly into two main categories:

Gradual wearing of certain regions of the face and flank of the cutting tool, and abrupt tool failure. Considering the more desirable case Œ the life of a cutting tool is therefore determined by the amount of wear that has occurred on the tool profile and which reduces the efficiency of cutting to an unacceptable level, or eventually causes tool failure. When the tool wear reaches an initially accepted amount, there are two options,

To resharpen the tool on a tool grinder, or

To replace the tool with a new one.

This second possibility applies in two cases,

When the resource for tool resharpening is exhausted. or

The tool does not allow for resharpening, e.g. in case of the indexable carbide inserts

Wear zones

Gradual wear occurs at three principal locations on a cutting tool. Accordingly, three main types of tool wear can be distinguished,

Crater wear

Flank wear

Corner wear

Crater wear: consists of a concave section on the tool face formed by the action of the chip sliding on the surface. Crater wear affects the mechanics of the process increasing the actual rake angle of the cutting tool and consequently, making cutting easier. At the same time, the crater wear weakens the tool wedge and increases the possibility for tool breakage. In general, crater wear is of a relatively small concern.

Flank wear: occurs on the tool flank as a result of friction between the machined surface of the workpiece and the tool flank. Flank wear appears in the form of so-called wear land and is measured by the width of this wear land, VB, Flank wear affects to the great extend the mechanics of cutting. Cutting forces increase significantly with flank wear. If the amount of flank wear exceeds some critical value (VB > 0.5~0.6 mm), the excessive cutting force may cause tool failure.

Corner wear: occurs on the tool corner. Can be considered as a part of the wear land and respectively flank wear since there is no distinguished boundary between the corner wear and flank wear land. We consider corner wear as a separate wear type because of its importance for the precision of machining. Corner wear actually shortens the cutting tool thus increasing gradually the dimension of machined surface and introducing a significant dimensional error in machining, which can reach values of about 0.03~0.05 mm.

Tool life

Tool wear is a time dependent process. As cutting proceeds, the amount of tool wear increases gradually. But tool wear must not be allowed to go beyond a certain limit in order to avoid tool failure. The most important wear type from the process point of view is the flank wear, therefore the parameter which has to be controlled is the width of flank wear land, VB. This parameter must not exceed an initially set safe limit, which is about 0.4 mm for carbide cutting tools. The safe limit is referred to as allowable wear land (wear criterion),

. The cutting time required for the cutting tool to develop a flank wear land of width is called tool life, T, a fundamental parameter in machining. The general relationship of VB versus cutting time is shown in the figure (so-called wear curve). Although the wear curve shown is for flank wear, a similar relationship occurs for other wear types. The figure shows also how to define the tool life T for a given wear criterion VBk

Parameters, which affect the rate of tool wear, are

Cutting conditions (cutting speed V, feed f, depth of cut d)

Cutting tool geometry (tool orthogonal rake angle)

Properties of work material

1.10 Surface finish

The machining processes generate a wide variety of surface textures. Surface texture consists of the repetitive and/or random deviations from the ideal smooth surface. These deviations are

Roughness: small, finely spaced surface irregularities (micro irregularities)

Waviness: surface irregularities of grater spacing (macro irregularities)

Lay: predominant direction of surface texture

Three main factors make the surface roughness the most important of these parameters:

Fatigue life: the service life of a component under cyclic stress (fatigue life) is much shorter if the surface roughness is high

Bearing properties: a perfectly smooth surface is not a good bearing because it cannot maintain a lubricating film.

Wear: high surface roughness will result in more intensive surface wear in friction.

Surface finish is evaluated quantitatively by the average roughness height, Ra

Roughness control

Factors, influencing surface roughness in machining are

Tool geometry (major cutting edge angle and tool corner radius),

Cutting conditions (cutting velocity and feed), and

Work material properties (hardness).

The influence of the other process parameters is outlined below:

Increasing the tool rake angle generally improves surface finish

Higher work material hardness results in better surface finish

Tool material has minor effect on surface finish.

Cutting fluids affect the surface finish changing cutting temperature and as a result the built-up edge formation.

1.11 Cutting fluids

Cutting fluid (coolant) is any liquid or gas that is applied to the chip and/or cutting tool to improve cutting performance. A very few cutting operations are performed dry, i.e., without the application of cutting fluids. Generally, it is essential that cutting fluids be applied to all machining operations.

Cutting fluids serve three principle functions:

To remove heat in cutting: the effective cooling action of the cutting fluid depends on the method of application, type of the cutting fluid, the fluid flow rate and pressure. The most effective cooling is provided by mist application combined with flooding. Application of fluids to the tool flank, especially under pressure, ensures better cooling that typical application to the chip but is less convenient.

To lubricate the chip-tool interface: cutting fluids penetrate the tool-chip interface improving lubrication between the chip and tool and reducing the friction forces and temperatures.

To wash away chips: this action is applicable to small, discontinuous chips only. Special devices are subsequently needed to separate chips from cutting fluids.

Methods of application

Manual application

Application of a fluid from a can manually by the operator. It is not acceptable even in job-shop situations except for tapping and some other operations where cutting speeds are very low and friction is a problem. In this case, cutting fluids are used as lubricants.

Flooding

In flooding, a steady stream of fluid is directed at the chip or tool-workpiece interface. Most machine tools are equipped with a recirculating system that incorporates filters for cleaning of cutting fluids. Cutting fluids are applied to the chip although better cooling is obtained by applying it to the flank face under pressure

Coolant-fed tooling

Some tools, especially drills for deep drilling, are provided with axial holes through the body of the tool so that the cutting fluid can be pumped directly to the tool cutting edge.

Mist applications

Fluid droplets suspended in air provide effective cooling by evaporation of the fluid. Mist application in general is not as effective as flooding, but can deliver cutting fluid to inaccessible areas that cannot be reached by conventional flooding.

Types of cutting fluid

Cutting Oils

Cutting oils are cutting fluids based on mineral or fatty oil mixtures. Chemical additives like sulphur improve oil lubricant capabilities. Areas of application depend on the properties of the particular oil but commonly, cutting oils are used for heavy cutting operations on tough steels.

Soluble Oils

The most common, cheap and effective form of cutting fluids consisting of oil droplets suspended in water in a typical ratio water to oil 30:1. Emulsifying agents are also added to promote stability of emulsion. For heavy-duty work, extreme pressure additives are used. Oil emulsions are typically used for aluminum and cooper alloys.

Chemical fluids

These cutting fluids consist of chemical diluted in water. They possess good flushing and cooling abilities. Tend to form more stable emulsions but may have harmful effects to the skin.

Environmental issues

Cutting fluids become contaminated with garbage, small chips, bacteria, etc., over time. Alternative ways of dealing with the problem of contamination are:

Replace the cutting fluid at least twice per month,

Machine without cutting fluids (dry cutting), Use a filtration system to continuously clean the cutting fluid.

Disposed cutting fluids must be collected and reclaimed. There are a number of methods of reclaiming cutting fluids removed from working area. Systems used range from simple settlement tanks to complex filtration and purification systems. Chips are emptied from the skips into a pulverizer and progress to centrifugal separators to become a scrap material. Neat oil after separation can be processed and returned, after cleaning and sterilizing to destroy bacteria.

1.12 Machinability

Machinability is a term indicating how the work material responds to the cutting process. In the most general case good machinability means that material is cut with good surface finish, long tool life, low force and power requirements, and low cost.

Machinability of different materials

Steels Leaded steels: lead acts as a solid lubricant in cutting to improve considerably machinability.

Resulphurized steels: sulphur forms inclusions that act as stress raisers in the chip formation zone thus increasing machinability.

Difficult-to-cut steels: a group of steels of low machinability, such as stainless steels, high manganese steels, precipitation-hardening steels.

Other metals

Aluminum: easy-to-cut material except for some cast aluminum alloys with silicon content that may be abrasive.

Cast iron: gray cast iron is generally easy-to-cut material, but some modifications and alloys are abrasive or very hard and may cause various problems in cutting.

Cooper-based alloys: easy to machine metals. Bronzes are more difficult to machine than brass.

Selection of cutting conditions

For each machining operation, a proper set of cutting conditions must be selected during the process planning. Decision must be made about all three elements of cutting conditions,

Depth of cut

Feed

Cutting speed

There are two types of machining operations:

Roughing operations: the primary objective of any roughing operation is to remove as much as possible material from the work piece for as short as possible machining time. In roughing operation, quality of machining is of a minor concern.

Finishing operations: the purpose of a finishing operation is to achieve the final shape, dimensional precision, and surface finish of the machined part. Here, the quality is of major importance. Selection of cutting conditions is made with respect to the type of machining operation. Cutting conditions should be decided in the order depth of cut - feed - cutting speed.

Somesh Sahu

UNIT II

TURNING MACHINES

2.1 Center Lathes

A lathe is a machine tool that rotates the work piece against a tool whose position it controls. The spindle is the part of the lathe that rotates. Various work holding attachments such as three jaw chucks, collets, and centers can be held in the spindle. The spindle is driven by an electric motor through a system of belt drives and gear trains. Spindle rotational speed is controlled by varying the geometry of the drive train.

The tailstock can be used to support the end of the workpiece with a center, or to hold tools for drilling, reaming, threading, or cutting tapers. It can be adjusted in position along the ways to accommodate different length workpieces. The tailstock barrel can be fed along the axis of rotation with the tailstock hand wheel.

The carriage controls and supports the cutting tool. It consists of:

A saddle that slides along the ways;

An apron that controls the feed mechanisms;

A cross slide that controls transverse motion of the tool (toward or away from the operator);

A tool compound that adjusts to permit angular tool movement; v a tool post that holds the cutting tools.

There are a number of different lathe designs, and some of the most popular are discussed here.

Centre lathe

The basic, simplest and most versatile lathe.

This machine tool is manually operated that is why it requires skilled operators. Suitable for low and medium production and for repair works.

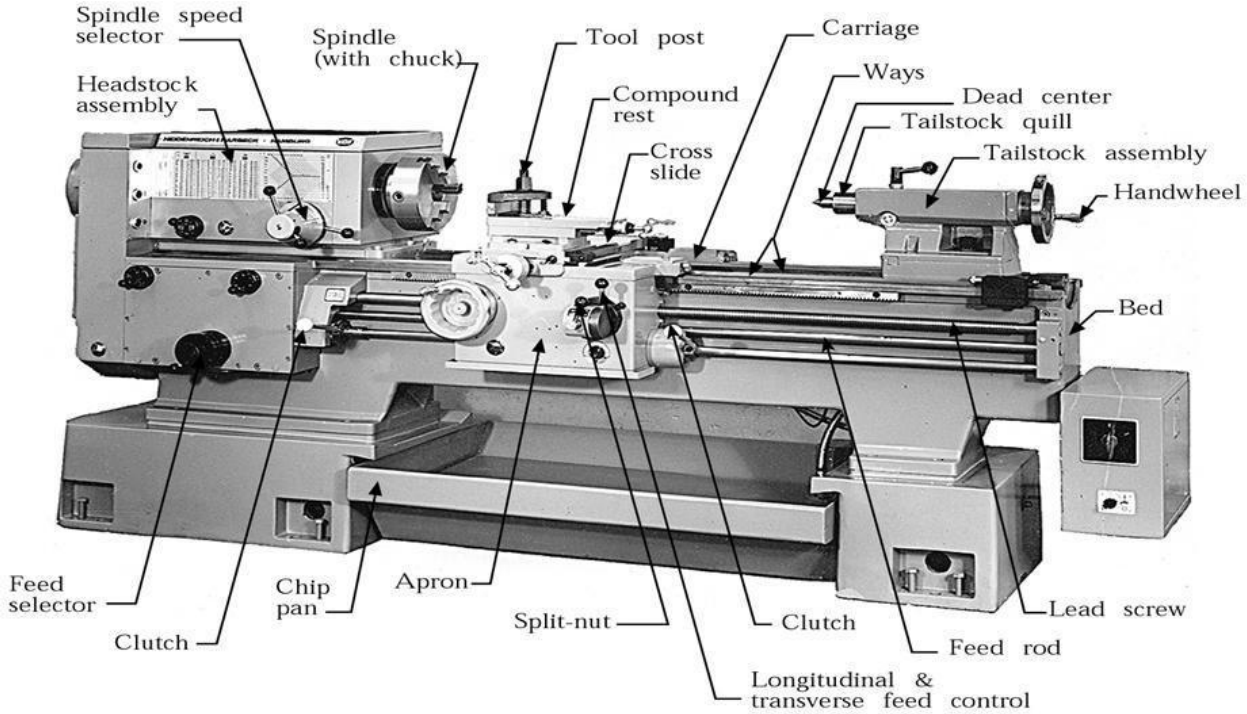
There are two tool feed mechanism in the engine lathes. These cause the cutting tool to move when engaged.

The lead screw will cause the apron and cutting tool to advance quickly. This is used for cutting threads, and for moving the tool quickly.

The feed rod will move the apron and cutting tool slowly forward. This is largely used for most of the turning operations.

Work is held in the lathe with a number of methods.

Between two centers. The work piece is driven by a device called a dog; the method is suitable for parts with high length-to-diameter ratio.

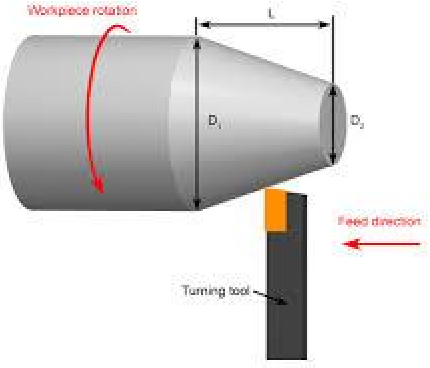


A 3 jaw self-centering chuck is used for most operations on cylindrical work parts. For parts with high length-to-diameter ratio the part is supported by center on the other end.

Collet consists of tubular bushing with longitudinal slits. Collets are used to grasp and hold bar stock. A collet of exact diameter is required to match any bar stock diameter.

A face plate is a device used to grasp parts with irregular shapes:

2.2 Taper turning methods



Copyright © 2007 CustomPartNet

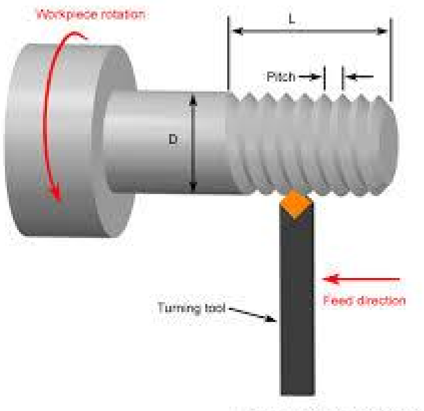
A taper is a conical shape. Tapers can be cut with lathes quite easily. There are some common methods for turning tapers on an center lathe,

Using a form tool: This type of tool is specifically designed for one cut, at a certain taper angle. The tool is plunged at one location, and never moved along the lathe slides. v Compound Slide

Method: The compound slide is set to travel at half of the taper angle. The tool is then fed across the work by hand, cutting the taper as it goes. v Off-Set Tail Stock: In this method the normal rotating part of the lathe still drives the workpiece (mounted between centres), but the centre at the tailstock is offset towards/away from the cutting tool. Then, as the cutting tool passes over, the part is cut in a conical shape. This method is limited to small tapers over long lengths.The tailstock offset h is defined by

$h = L\sin\alpha$, where L is the length of work piece, and α is the half of the taper angle.

2.3 Thread cutting methods



Copyright © 2007 CustomPartNet

Different possibilities are available to produce a thread on a lathe. Threads are cut using lathes by advancing the cutting tool at a feed exactly equal to the thread pitch. The single-point cutting tool cuts in a helical band, which is actually a thread. The procedure calls for correct settings of the machine, and also that the helix be restarted at the same location each time if multiple passes are required to cut the entire depth of thread. The tool point must be ground so that it has the same profile as the thread to be cut.

Another possibility is to cut threads by means of a thread die (external threads), or a tap (internal threads). These operations are generally performed manually for small thread diameters.

2.4 Special Attachments

Unless a workpiece has a taper machined onto it which perfectly matches the internal taper in the spindle, or has threads which perfectly match the external threads on the spindle (two conditions which rarely exist), an accessory must be used to mount a workpiece to the spindle.

A workpiece may be bolted or screwed to a faceplate, a large, flat disk that mounts to the spindle. In the alternative, faceplate dogs may be used to secure the work to the faceplate.

A workpiece may be mounted on a mandrel, or circular work clamped in a three- or four-jaw chuck. For irregular shaped workpieces it is usual to use a four jaw (independent moving jaws) chuck. These holding devices mount directly to the Lathe headstock spindle.

In precision work, and in some classes of repetition work, cylindrical workpieces are usually held in a collet inserted into the spindle and secured either by a draw-bar, or by a collet closing cap on the spindle. Suitable collets may also be used to mount square or hexagonal workpieces. In precision tool making work such collets are usually of the draw-in variety, where, as the collet is tightened, the workpiece moves slightly back into the headstock, whereas for most repetition work the dead length variety is preferred, as this ensures that the position of the workpiece does not move as the collet is tightened.

A soft workpiece (e.g., wood) may be pinched between centers by using a spur drive at the headstock, which bites into the wood and imparts torque to it.

2.5 Machining time

Machining time is the time when a machine is actually processing something. Generally, machining time is the term used when there is a reduction in material or removing some undesirable parts of a material. For example, in a drill press, machining time is when the cutting edge is actually moving forward and making a hole. Machine time is used in other situations, such as when a machine installs screws in a case automatically.

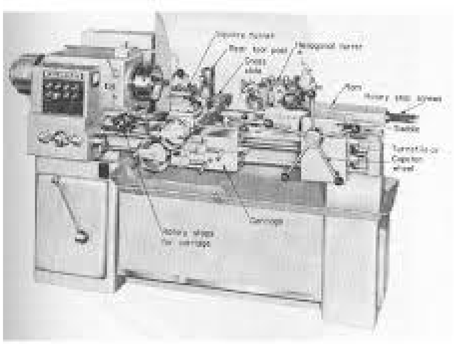
One of the important aspects in manufacturing calculation is how to find and calculate the machining time in a machining operation. Generally, machining is family of processes or operations in which excess material is removed from a starting work piece by a sharp cutting tool so the remaining part has the desired geometry and the required shape. The most common machining operations can be classified into four types: turning, milling, drilling and lathe work.

Calculate Time for Turning

$$\text{Time for Turning} = \frac{\text{Length of the job to be turned}}{\text{Feed/Rev.} \times \text{r.p.m.}} \text{ min.}$$

2.6 Capstan versus turret

Capstan Lathe



Turret Lathe

The term "capstan lathe" overlaps in sense with the term "turret lathe" to a large extent. In many times and places, it has been understood to be synonymous with "turret lathe". In other times and places it has been held in technical contradistinction to "turret lathe", with the difference being in whether the turret's slide is fixed to the bed (ram-type turret) or slides on the bed's ways (saddle-type turret). The difference in terminology is mostly a matter of United Kingdom and Common wealth usage versus United States usage. American usage tends to call them all "turret lathes".

The word "capstan" could logically seem to refer to the turret itself, and to have been inspired by the nautical capstan. A lathe turret with tools mounted in it can very much resemble a nautical capstan full of handspikes. This interpretation would lead Americans to treat "capstan" as a synonym of "turret" and "capstan lathe" as a synonym of "turret lathe". However, the multi-spoked handles that the operator uses to advance the slide are also called capstans, and they themselves also resemble the nautical capstan.

No distinction between "turret lathe" and "capstan lathe" persists upon translation from English into other languages. Most translations involve the term "revolver", and serve to translate either of the English terms.

The words "turret" and "tower", the former being a diminutive of the latter, come ultimately from the Latin "turris", which means "tower", and the use of "turret" both to refer to lathe turrets and to refer to gun turrets seems certainly to have been inspired by its earlier connection to the turrets of fortified buildings and to siege towers. The history of the rook in chess is connected to the same history, with the French word for rook, tour, meaning "tower".

It is an interesting coincidence that the word "tour" in French can mean both "lathe" and "tower", with the first sense coming ultimately from Latin "tornus", "lathe", and the second sense coming ultimately from Latin "turris", "tower". "Tour revolver", "tour tourelle", and "tour tourelle revolver" are various ways to say "turret lathe" in French.

2.7 Semi-automatic

Sometimes machines similar to those above, but with power feeds and automatic turret-indexing at the end of the return stroke, are called "semi-automatic turret lathes". This nomenclature distinction is blurry and not consistently observed. The term "turret lathe" encompasses them all. During the 1860s, when semi-automatic turret lathes were developed, they were sometimes called "automatic". What we today would call "automatics", that is, fully automatic machines, had not been developed yet. During that era both manual and semi-automatic turret lathes were sometimes called "screw machines", although we today reserve that term for fully automatic machines.

2.8 Automatic

During the 1870s through 1890s, the mechanically automated "automatic" turret lathe was developed and disseminated. These machines can execute many part-cutting cycles without human intervention. Thus the duties of the operator, which were already greatly reduced by the manual turret lathe, were even further reduced, and productivity increased. These machines use cams to automate the sliding and indexing of the turret and the opening and closing of the chuck. Thus, they execute the part-cutting cycle somewhat analogously to the way in which an elaborate cuckoo clock performs an automated theater show. Small- to medium-sized automatic turret lathes are usually called "screw machines" or "automatic screw machines", while larger ones are usually called "automatic chucking lathes", "automatic chuckers", or "chuckers".

UNIT III

SHAPER, MILLING AND GEAR CUTTING MACHINES

3.1 Shapers

Shaping is performed on a machine tool called a shaper. The major components of a shaper are the ram, which has the tool post with cutting tool mounted on its face, and a worktable, which holds the part and accomplishes the feed motion.

A shaper is a type of machine tool that uses linear relative motion between the workpiece and a single-point cutting tool tomachine a linear toolpath. Its cut is analogous to that of a lathe, except that it is (archetypally) linear instead of helical. (Adding axes of motion can yield helical toolpaths, as also done in helical planing.) A shaper is analogous to a planer, but smaller, and with the cutter riding a ram that moves above a stationary workpiece, rather than the entire workpiece moving beneath the cutter. The ram is moved back and forth typically by a crank inside the column; hydraulically actuated shapers also exist.

3.2 Types of Shapers

Shapers are mainly classified as standard, draw-cut, horizontal, universal, vertical, geared, crank, hydraulic, contour and traveling head.[1] The horizontal arrangement is the most common. Vertical shapers are generally fitted with a rotary table to enable curved surfaces to be machined (same idea as in helical planing). The vertical shaper is essentially the same thing as a slotter (slotting machine), although technically a distinction can be made if one defines a true vertical shaper as a machine whose slide can be moved from the vertical. A slotter is fixed in the vertical plane.

Small shapers have been successfully made to operate by hand power. As size increases, the mass of the machine and its power requirements increase, and it becomes necessary to use a motor or other supply of mechanical power. This motor drives a mechanical arrangement (using a pinion gear, bull gear, and crank, or a chain over sprockets) or a hydraulic motor that supplies the necessary movement via hydraulic cylinders.

The workpiece mounts on a rigid, box-shaped table in front of the machine. The height of the table can be adjusted to suit this workpiece, and the table can traverse sideways underneath the reciprocating tool, which is mounted on the ram. Table motion may be controlled manually, but is usually advanced by an automatic feed mechanism acting on the feedscrew.

The ram slides back and forth above the work. At the front end of the ram is a vertical tool slide that may be adjusted to either side of the vertical plane along the stroke axis. This tool-slide holds the *clapper box* and toolpost, from which the tool can be positioned to cut a straight, flat surface on the top of the workpiece. The tool-slide permits feeding the tool downwards to deepen a cut. This adjustability, coupled with the use of specialized cutters and toolholders, enable the operator to cut internal and external gear tooth profiles, splines, dovetails, and keyways.

The most common use is to machine straight, flat surfaces, but with ingenuity and some accessories a wide range of work can be done. Other examples of its use are:

- Keyways in the boss of a pulley or gear can be machined without resorting to a dedicated broaching setup.
- Dovetail slides
- Internal splines and gear teeth.
- Keyway, spline, and gear tooth cutting in blind holes
- Cam drums with toolpaths of the type that in CNC milling terms would require 4- or 5-axis contouring or turn-mill cylindrical interpolation
- It is even possible to obviate wire EDM work in some cases. Starting from a drilled or cored hole, a shaper with a boring-bar type tool can cut internal features that don't lend themselves to milling or boring (such as irregularly shaped holes with tight corners).

3.3 Drilling and Reaming

Drilling and reaming operations

Drilling operation

Drilling is used to drill a round blind or through hole in a solid material. If the hole is larger than ~30 mm, its a good idea to drill a smaller pilot hole before core drilling the final one. For holes larger than ~50 mm, three-step drilling is recommended; v Core drilling is used to increase the diameter of an existing hole; v Step drilling is used to drill a stepped (multi-diameter) hole in a solid material;

Counterboring provides a stepped hole again but with flat and perpendicular relative to hole axis face. The hole is used to seat internal hexagonal bolt heads;

Countersinking is similar to counterboring, except that the step is conical for flat head screws:

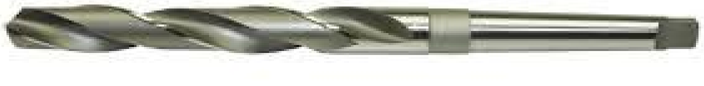
Reaming provides a better tolerance and surface finish to an initially drilled hole. Reaming slightly increases the hole diameter. The tool is called reamer;

Center drilling is used to drill a starting hole to precisely define the location for subsequent drilling. The tool is called center drill. A center drill has a thick shaft and very short flutes. It is therefore very stiff and will not walk as the hole is getting started;

Gun drilling is a specific operation to drill holes with very large length-to-diameter ratio up to L/D ~300. There are several modifications of this operation but in all cases cutting fluid is delivered directly to the cutting zone internally through the drill to cool and lubricate the cutting edges, and to remove the chips (see Section 5.6 Cutting Fluids);

Drills and Reamers

Reamer



Twist drill

The twist drill does most of the cutting with the tip of the bit. It has two flutes to carry the chips up from the cutting edges to the top of the hole where they are cast off. The standard drill geometry

The typical helix angle of a general purpose twist drill is 18~30 degree, while the point angle (which equals two times the major cutting edge angle, see page 101) for the same drill is 118deg.

Some standard drill types are,

straight shank: this type has a cylindrical shank and is held in a chuck;

taper shank: his type is held directly in the drilling machine spindle.

Reamers

The reamer has similar geometry. The difference in geometry between a reamer and a twist drill are:

The reamer contains four to eight straight or helical flutes, respectively cutting edges.

The tip is very short and does not contain any cutting edges.

3.4 Boring

Boring is a process of producing circular internal profiles on a hole made by drilling or another process. It uses single point cutting tool called a boring bar. In boring, the boring bar can be rotated, or the workpart can be rotated. Machine tools which rotate the boring bar against a stationary workpiece are called boring machines (also boring mills). Boring can be accomplished on a turning machine with a stationary boring bar positioned in the tool post and rotating workpiece held in the lathe chuck as illustrated in the figure. In this section, we will consider only boring on boring machines.

Vertical Boring

Boring machines

Boring machines can be horizontal or vertical according to the orientation of the axis of rotation of the machine spindle. In horizontal boring operation, boring bar is mounted in a tool slide, which position is adjusted relative to the spindle face plate to machine different diameters. The boring bar must be supported on the other end when boring long and small-diameter holes. A vertical boring mill is used for large, heavy work parts with diameters up to 12 m. The typical boring mill can position and feed several cutting tools simultaneously. The work part is mounted on a rotating worktable.

Cutting tool for boring

The typical boring bar is shown in the figure. When boring with a rotating tool, size is controlled by changing the radial position of the tool slide, which holds the boring bar, with respect to the spindle axis of rotation. For finishing machining, the boring bar is additionally mounted in an adjustable boring head for more precise control of the bar radial position.

3.5 Tapping

A tap cuts a thread on the inside surface of a hole, creating a female surface which functions like a nut. The three taps in the image illustrate the basic types commonly used by most machinists:

Taps

Bottoming tap or plug taps

The tap illustrated in the top of the image has a continuous cutting edge with almost no taper — between 1 and 1.5 threads of taper is typical. This feature enables a bottoming tap to cut threads to the bottom of a blind hole. A bottoming tap is usually used to cut threads in a hole that has already been partially threaded using one of the more tapered types of tap; the tapered end ("tap chamfer") of a bottoming tap is too short to successfully start into an unthreaded hole. In the US, they are commonly known as bottoming taps, but in Australia and Britain they are also known as plug taps.

Intermediate tap, second tap, or plug tap

The tap illustrated in the middle of the image has tapered cutting edges, which assist in aligning and starting the tap into an untapped hole. The number of tapered threads typically ranges from 3 to 5.Plug taps are the most commonly used type of tap.[citation needed] In the US, they are commonly known as plug taps, whereas in Australia and Britain they are commonly known as second taps.

3.6 Milling

Milling is a process of producing flat and complex shapes with the use of multi-tooth cutting tool, which is called a milling cutter and the cutting edges are called teeth. The axis of rotation of the cutting tool is perpendicular to the direction of feed, either parallel or perpendicular to the machined surface. The machine tool that traditionally performs this operation is a milling machine. Milling is an interrupted cutting operation: the teeth of the milling cutter enter and exit the work during each revolution. This interrupted cutting action subjects the teeth to a cycle of impact force and thermal shock on every rotation. The tool material and cutter geometry must be designed to withstand these conditions. Cutting fluids are essential for most milling operations. Three types of feed in milling can be identified:

Feed per tooth: the basic parameter in milling equivalent to the feed in turning.

Feed per tooth is selected with regard to the surface finish and dimensional accuracy required. Feeds per tooth are in the range of 0.05~0.5 mm/tooth, lower feeds are for finishing cuts; feed per revolution: it determines the amount of material cut per one full revolution of the milling cutter. Feed per revolution is calculated as fr = fz being the number of the cutter's teeth;

Feed per minute fm: Feed per minute is calculated taking into account the rotational speed

N and number of the cutter's teeth z, fm = fzN = frN

Feed per minute is used to adjust the feed change gears.

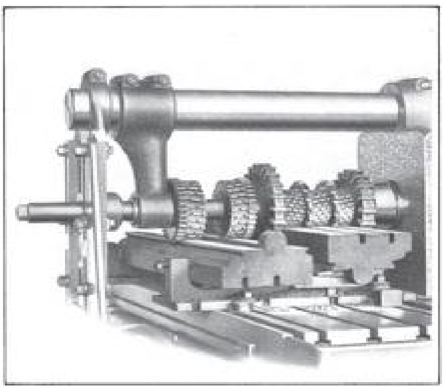
Three types of feed in milling can be identified:

Feed per tooth fz: the basic parameter in milling equivalent to the feed in turning.

Feed per tooth is selected with regard to the surface finish and dimensional accuracy required (see Section 5.10 Selection of Cutting Conditions). Feeds per tooth are in the range of 0.05~0.5 mm/tooth, lower feeds are for finishing cuts; feed per revolution fr: it determines the amount of material cut per one full revolution of the milling cutter. Feed per revolution is calculated as

fr = fz ,z being the number of the cutter's teeth;

Feed per minute fm: Feed per minute is calculated taking into account the rotational speed N and number of the cutter's teeth z, fm = fzN = fr,NFeed per minute is used to adjust the feed change gears. In down milling, the cutting force is directed into the work table, which allows thinner workparts tobe machined. Better surface finish is obtained but the stress load on the teeth is abrupt, which may damage the cutter.In up milling, the cutting force tend to lift the workpiece. The work conditions for the cutter are more favourable. Because the cutter does not start to cut when it makes contact (cutting at zero cut is impossible), the surface has a natural waviness.



Milling Operations

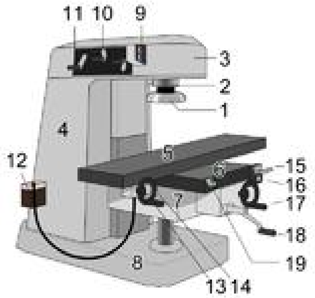
Owing to the variety of shapes possible and its high production rates, milling is one of the most versatile and widely used machining operations. The geometric form created by milling fall into three major groups: Plane surfaces: the surface is linear in all three dimensions. The simplest and most convenient type of surface;

Two-dimensional surfaces: the shape of the surface changes in the direction of two of the axes and is linear along the third axis. Examples include cams;

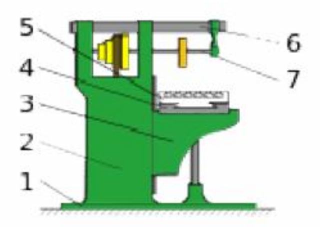
Three-dimensional surfaces: the shape of the surface changes in all three directions.

Examples include die cavities, gas turbine blades, propellers, casting patterns, etc.

Milling machines



Vertical milling machine



Horizontal milling machine

The conventional milling machines provide a primary rotating motion for the cutter held in the spindle, and a linear feed motion for the workpiece, which is fastened onto the worktable. Milling machines for machining of complex shapes usually provide both a rotating primary motion and a curvilinear feed motion for the cutter in the spindle with a stationary workpiece. Various machine designs are available for various milling operations. In this section we discuss only the most popular ones, classified into the following types:

Column-and-knee milling machines; v Bed type milling machines;

Machining centers.

Column-and-knee milling machines

The column-and-knee milling machines are the basic machine tool for milling. The name comes from the fact that this machine has two principal components, a column that supports the spindle, and a knee that supports the work table. There are two different types of column-and-knee milling machines according to position of the spindle axis:

horizontal, and vertical.

Milling cutters

Brazed cutters: Very limited numbers of cutters (mainly face mills) are made with brazed carbide inserts. This design is largely replaced by mechanically attached cutters.

Mechanically attached cutters: The vast majority of cutters are in this category. Carbide inserts are either clamped or pin locked to the body of the milling cutter.

Classification of milling cutters may also be associated with the various milling operations

3.7 Gear

Gears can be manufactured by most of manufacturing processes discussed so far (casting, forging, extrusion, powder metallurgy, blanking). But as a rule, machining is applied to achieve the final dimensions, shape and surface finish in the gear. The initial operations that produce a semi finishing part ready for gear machining as referred to as blanking operations; the starting product in gear machining is called a gear blank.

Two principal methods of gear manufacturing include

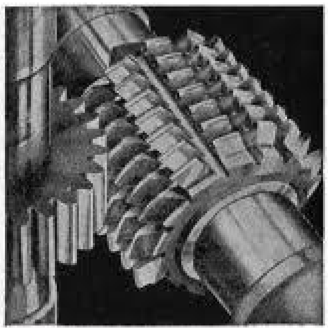
Gear forming, and Gear generation.

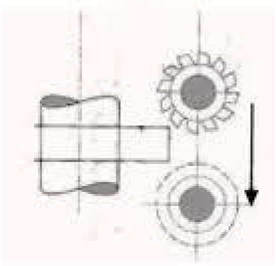
Each method includes a number of machining processes, the major of them included in this section.

Gear forming

In gear form cutting, the cutting edge of the cutting tool has a shape identical with the shape of the space between the gear teeth.

Two machining operations, milling and broaching can be employed to form cut gear teeth





3.8 Gear milling

In form milling, the cutter called a form cutter travels axially along the length of the gear tooth at the appropriate depth to produce the gear tooth. After each tooth is cut, the cutter is withdrawn, the gear blank is rotated (indexed), and the cutter proceeds to cut another tooth. The process continues until all teeth are cut.

Each cutter is designed to cut a range of tooth numbers. The precision of the form-cut tooth profile depends on the accuracy of the cutter and the machine and its stiffness. In form milling, indexing of the gear blank is required to cut all the teeth. Indexing is the process of evenly dividing the circumference of a gear blank into equally spaced divisions. The index head of the indexing fixture is used for this purpose.

The index fixture consists of an index head (also dividing head, gear cutting attachment) and footstock, which is similar to the tailstock of a lathe. The index head and footstock attach to the worktable of the milling machine. An index plate containing graduations is used to control the rotation of the index head spindle. Gear blanks are held between centers by the index head spindle and footstock. Workpieces may also be held in a chuck mounted to the index head spindle or may be fitted directly into the taper spindle recess of some indexing fixtures.

3.9 Gear hobbing

Gear hobbing is a machining process in which gear teeth are progressively generated by a series of cuts with a helical cutting tool (hob). All motions in hobbing are rotary, and the hob and gear blank rotate continuously as in two gears meshing until all teeth are cut when bobbing a spur gear, the angle between the hob and gear blank axes is 90° minus the lead angle at the hob threads. For helical gears, the hob is set so that the helix angle of the hob is parallel with the tooth direction of the gear being cut. Additional movement along the tooth length is necessary in order to cut the whole tooth length: The action of the hobbing machine (also gear hobber) is shown in the figures. The cutting of a gear by means of a hob is a continuous operation. The hob and the gear blank are connected by a proper gearing so that they rotate in mesh. To start cutting a gear, the rotating hob is fed inward until the proper setting for tooth depth is achieved, then cutting continues until the entire gear is finished.

The gear hob is a formed tooth milling cutter with helical teeth arranged like the thread on a screw. These teeth are fluted to produce the required cutting edges.

3.10 Shaping with a pinion-shaped cutter

This modification of the gear shaping process is defined as a process for generating gear teeth by a rotating and reciprocating pinion-shaped cutter:

The cutter axis is parallel to the gear axis. The cutter rotates slowly in timed relationship with the gear blank at the same pitch-cycle velocity, with an axial primary reciprocating motion; to produce the gear teeth. A train of gears provides the required relative motion between the cutter shaft and the gear-blank shaft. Cutting may take place either at the down stroke or upstroke of the machine. Because the clearance required for cutter travel is small, gear shaping is suitable for gears that are located close to obstructing surfaces such as flanges. The tool is called gear cutter and resembles in shape the mating gear from the conjugate gear pair, the other gear being the blank.

Gear shaping is one of the most versatile of all gear cutting operations used to produce internal gears, external gears, and integral gear-pinion arrangements. Advantages of gear shaping with pinion-shaped cutter are the high dimensional accuracy achieved and the not too expensive tool. The process is applied for finishing operation in all types of production rates.

3.11 Finishing operations

As produced by any of the process described, the surface finish and dimensional accuracy may not be accurate enough for certain applications. Several finishing operations are available, including the conventional process of shaving, and a number of abrasive operations, including grinding, honing, and lapping.

UNIT IV

ABRASIVE PROCESS AND BROACHING

4.1 Abrasive Processes

Abrasive machining processes can be divided into two categories based on how the grains are applied to the workpiece.

In bonded abrasive processes, the particles are held together within a matrix, and their combined shape determines the geometry of the finished workpiece. For example, in grinding the particles are bonded together in a wheel. As the grinding wheel is fed into the part, its shape is transferred onto the workpiece.

In loose abrasive processes, there is no structure connecting the grains. They may be applied without lubrication as dry powder, or they may be mixed with a lubricant to form a slurry. Since the grains can move independently, they must be forced into the workpiece with another object like a polishing cloth or a lapping plate.

Common abrasive processes are listed below.

Fixed (bonded) abrasive processes

- Grinding
- Honing, superfinishing
- Tape finishing, abrasive belt machining
- Buffing, brushing
- Abrasive sawing, Diamond wire cutting, Wire saw
- Sanding

Loose abrasive processes

- Polishing
- Lapping
- Abrasive flow machining (AFM)
- Hydro-erosive grinding
- Water-jet cutting
- Abrasive blasting
- Mass finishing,

4.2 Grinding Wheels

A grinding wheel is an expendable wheel that is composed of an abrasive compound used for various grinding (abrasive cutting) and abrasive machining operations. They are used in grinding machines.

The wheels are generally made from a matrix of coarse particles pressed and bonded together to form a solid, circular shape. Various profiles and cross sections are available depending on the

intended usage for the wheel. They may also be made from a solid steel or aluminium disc with particles bonded to the surface.

The manufacture of these wheels is a precise and tightly controlled process, due not only to the inherent safety risks of a spinning disc, but also the composition and uniformity required to prevent that disc from exploding due to the high stresses produced on rotation.

There are five characteristics of a cutting wheel: material, grain size, wheel grade, grain spacing, and bond type. They will be indicated by codes on the wheel's label.

Abrasive Grain, the actual abrasive, is selected according to the hardness of the material being cut.

- Aluminum Oxide (A)
- Silicon Carbide (S)
- Ceramic (C)
- Diamond (D, MD, SD)
- Cubic Boron Nitride (B)

Grinding wheels with diamond or Cubic Boron Nitride (CBN) grains are called superabrasives. Grinding wheels with Aluminum Oxide (corundum), Silicon Carbide or Ceramic grains are called conventional abrasives.

Grain size, from 8 (coarsest) 1200 (finest), determines the physical size of the abrasive grains in the wheel. A larger grain will cut freely, allowing fast cutting but poor surface finish. Ultra-fine grain sizes are for precision finish work.

Wheel grade, from A (soft) to Z (hard), determines how tightly the bond holds the abrasive. Grade affects almost all considerations of grinding, such as wheel speed, coolant flow, maximum and minimum feed rates, and grinding depth.

Grain spacing, or structure, from 1 (densest) to 16 (least dense). Density is the ratio of bond and abrasive to air space. A less-dense wheel will cut freely, and has a large effect on surface finish. It is also able to take a deeper or wider cut with less coolant, as the chip clearance on the wheel is greater.

Wheel bond, how the wheel holds the abrasives, affects finish, coolant, and minimum/maximum wheel speed.

- Vitrified (V)
- Resinoid (B)
- Silicate (S)

- Shellac (E)
- Rubber (R)
- Metal (M)
- Oxychloride (O)

4.3 Types of Grinding Processes

Straight wheel

Straight wheel

To the right is an image of a straight wheel. These are by far the most common style of wheel and can be found on bench or pedestal grinders. They are used on the periphery only and therefore produce a slightly concave surface (*hollow ground*) on the part. This can be used to advantage on many tools such as chisels.

Straight Wheels are generally used for cylindrical, centreless, and surface grinding operations. Wheels of this form vary greatly in size, the diameter and width of face naturally depending upon the class of work for which is used and the size and power of the grinding machine.

Cylinder or wheel ring

Cylinder wheels provide a long, wide surface with no center mounting support (hollow). They can be very large, up to 12" in width. They are used only in vertical or horizontal spindle grinders. Cylinder or wheel ring is used for producing flat surfaces, the grinding being done with the end face of the wheel.

Tapered wheel

A straight wheel that tapers outward towards the center of the wheel. This arrangement is stronger than straight wheels and can accept higher lateral loads. Tapered face straight wheel is primarily used for grinding thread, gear teeth etc.

Straight cup

Straight cup wheels are an alternative to cup wheels in tool and cutter grinders, where having an additional radial grinding surface is beneficial.

Dish cup

A very shallow cup-style grinding wheel. The thinness allows grinding in slots and crevices. It is used primarily in cutter grinding and jig grinding.

Saucer wheel

A special grinding profile that is used to grind milling cutters and twist drills. It is most common in non-machining areas, as saw filers use saucer wheels in the maintenance of saw blades.

Diamond wheels

Diamond wheel

Diamond wheels are grinding wheels with industrial diamonds bonded to the periphery.

They are used for grinding extremely hard materials such as carbide cutting tips, gemstones or concrete. The saw pictured to the right is a slitting saw and is designed for slicing hard materials, typically gemstones.

Mounted points

Mounted points are small grinding wheels bonded onto a mandrel. Diamond mounted points are tiny diamond rasps for use in a jig grinder doing profiling work in hard material. Resin and vitrified bonded mounted points with conventional grains are used for deburring applications, especially in the foundry industry.

Cut off wheels

Cut off wheels, also known as *parting wheels*, are self-sharpening wheels that are thin in width and often have radial fibres reinforcing them. They are often used in the construction industry for cutting reinforcement bars (rebar), protruding bolts or anything that needs quick removal or trimming. Most handymen would recognise an angle grinder and the discs they use.

4.4 Cylindrical grinding

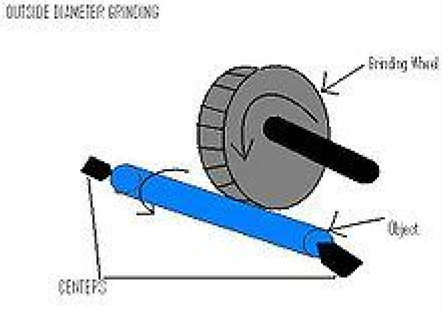
The cylindrical grinder is a type of grinding machine used to shape the outside of an object. The cylindrical grinder can work on a variety of shapes; however the object must have a central axis of rotation. This includes but is not limited to such shapes as a cylinder, an ellipse, a cam, or a crankshaft.

Cylindrical grinding is defined as having four essential actions:

1. The work (object) must be constantly rotating
2. The grinding wheel must be constantly rotating
3. The grinding wheel is fed towards and away from the work
4. Either the work or the grinding wheel is traversed with respect to the other.

While the majority of cylindrical grinders employ all four movements, there are grinders that only employ three of the four actions.

There are five different types of cylindrical grinding: outside diameter (OD) grinding, inside diameter (ID) grinding, plunge grinding, creep feed grinding, and centerless grinding.



A basic overview of Outside Diameter Cylindrical Grinding. The Curved Arrows refer to direction of rotation

4.5 Outside Diameter Grinding

OD grinding is grinding occurring on external surface of an object between the centers. The centers are end units with a point that allow the object to be rotated. The grinding wheel is also being rotated in the same direction when it comes in contact with the object. This effectively means the two surfaces will be moving opposite directions when contact is made which allows for a smoother operation and less chance of a jam up.

Plunge grinding

A form of OD grinding, however the major difference is that the grinding wheel makes continuous contact with a single point of the object instead of traversing the object.

Creep feed grinding

Creep Feed is a form of grinding where a full depth of cut is removed in a single pass of the wheel. Successful operation of this technique can reduce manufacturing time by 50%, but often the grinding machine being used must be designed specifically for this purpose. This form occurs in both cylindrical and

Surface Grinding

Surface grinding is used to produce a smooth finish on flat surfaces. It is a widely used abrasive machining process in which a spinning wheel covered in rough particles (grinding wheel) cuts chips of metallic or nonmetallic substance from a workpiece, making a face of it flat or smooth.

Surface grinding is the most common of the grinding operations. It is a finishing process that uses a rotating abrasive wheel to smooth the flat surface of metallic or nonmetallic materials to give them a more refined look or to attain a desired surface for a functional purpose.

The surface grinder is composed of an abrasive wheel, a workholding device known as a chuck, and a reciprocating or rotary table. The chuck holds the material in place while it is being worked on. It can do this one of two ways: ferromagnetic pieces are held in place by a magnetic chuck, while non-ferromagnetic and nonmetallic pieces are held in place by vacuum or mechanical means. A machine vise (made from ferromagnetic steel or cast iron) placed on the magnetic chuck can be used to hold non-ferromagnetic workpieces if only a magnetic chuck is available.

Factors to consider in surface grinding are the material of the grinding wheel and the material of the piece being worked on.

Typical workpiece materials include cast iron and mild steel. These two materials don't tend to clog the grinding wheel while being processed. Other materials are aluminum, stainless steel, brass and some plastics. When grinding at high temperatures, the material tends to become weakened and is more inclined to corrode. This can also result in a loss of magnetism in materials where this is applicable.

The grinding wheel is not limited to a cylindrical shape and can have a myriad of options that are useful in transferring different geometries to the object being worked on. Straight wheels can be dressed by the operator to produce custom geometries. When surface grinding an object, one must keep in mind that the shape of the wheel will be transferred to the material of the object like a mirror image.

Spark out is a term used when precision values are sought and literally means "until the sparks are out (no more)". It involves passing the workpiece under the wheel, without resetting the depth of cut, more than once and generally multiple times. This ensures that any inconsistencies in the machine or workpiece are eliminated.

A surface grinder is a machine tool used to provide precision ground surfaces, either to a critical size or for the surface finish.

The typical precision of a surface grinder depends on the type and usage, however +/- 0.002 mm (+/- 0.0001") should be achievable on most surface grinders.

The machine consists of a table that traverses both longitudinally and across the face of the wheel. The longitudinal feed is usually powered by hydraulics, as may the cross feed, however any mixture of hand, electrical or hydraulic may be used depending on the ultimate usage of the machine (i.e.: production, workshop, cost). The grinding wheel rotates in the spindle head and is also adjustable for height, by any of the methods described previously. Modern surface grinders are semi-automated, depth of cut and spark-out may be preset as to the number of passes and, once set up, the machining process requires very little operator intervention.

Depending on the workpiece material, the work is generally held by the use of a magnetic chuck. This may be either an electromagnetic chuck, or a manually operated, permanent magnet type chuck; both types are shown in the first image.

The machine has provision for the application of coolant as well as the extraction of metal dust (metal and grinding particles).

Types of surface grinders

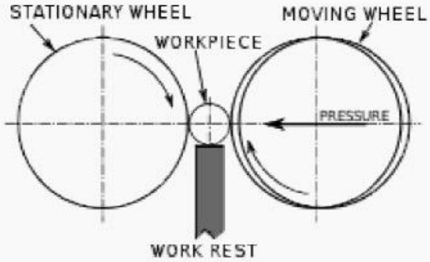
Horizontal-spindle (peripheral) surface grinders**.** The periphery (flat edge) of the wheel is in contact with the workpiece, producing the flat surface. Peripheral grinding is used in high-precision work on simple flat surfaces; tapers or angled surfaces; slots; flat surfaces next to shoulders; recessed surfaces; and profiles.

Vertical-spindle (wheel-face) grinders**.** The face of a wheel (cup, cylinder, disc, or segmental wheel) is used on the flat surface. Wheel-face grinding is often used for fast material removal, but some machines can accomplish high-precision work. The workpiece is held on a reciprocating table, which can be varied according to the task, or a rotary-table machine, with continuous or indexed rotation. Indexing allows loading or unloading one station while grinding operations are being performed on another.

Disc grinders and double-disc grinders. Disc grinding is similar to surface grinding, but with a larger contact area between disc and workpiece. Disc grinders are available in both vertical and horizontal spindle types. Double disc grinders work both sides of a workpiece simultaneously. Disc grinders are capable of achieving especially fine tolerances.

4.6 Centerless grinding

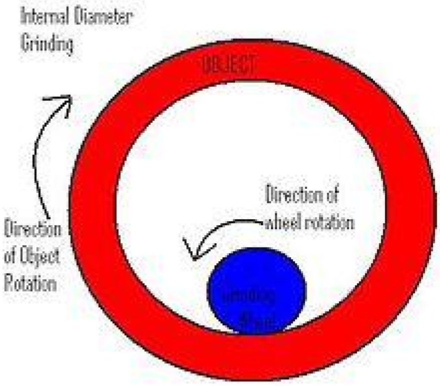
Centerless cylindrical grinder



A schematic of the centerless grinding process.

Centerless grinding is a form of grinding where there is no collet or pair of centers holding the object in place. Instead, there is a regulating wheel positioned on the opposite side of the object to the grinding wheel. A work rest keeps the object at the appropriate height but has no bearing on its rotary speed. The workblade is angled slightly towards the regulating wheel, with the workpiece centerline above the centerlines of the regulating and grinding wheel; this means that high spots do not tend to generate corresponding opposite low spots, and hence the roundness of parts can be improved. Centerless grinding is much easier to combine with automatic loading procedures than centered grinding; throughfeed grinding, where the regulating wheel is held at a slight angle to the part so that there is a force feeding the part through the grinder, is particularly efficient.

4.7 Internal Grinding



A basic overview of Internal Diameter Cylindrical Grinding. The Curved Arrows refer to direction of rotation.

ID grinding is grinding occurring on the inside of an object. The grinding wheel is always smaller than the width of the object. The object is held in place by a collet, which also rotates the object in place. Just as with OD grinding, the grinding wheel and the object rotated in opposite directions giving reversed direction contact of the two surfaces where the grinding occurs.

4.8 Concepts of surface Integrity

Surface integrity is the surface condition of a workpiece after being modified by a manufacturing process. The surface integrity of a workpiece or item changes the material's properties. The consequences of changes to surface integrity are a mechanical engineering design problem, but the preservation of those properties are a manufacturing consideration.

Surface integrity can have a great impact on a parts function; for example, Inconel 718 can have a fatigue limit as high as 540 MPa (78,000 psi) after a gentlegrinding or as low as 150 MPa (22,000 psi) after electrical discharge machining (EDM).

There are two aspects to surface integrity: topography characteristics and surface layer characteristics. The topography is made up of surface roughness, waviness, errors of form, and flaws. The surface layer characteristics that can change through processing are: plastic deformation, residual stresses, cracks, hardness, overaging, phase changes, recrystallization, intergranular attack, and hydrogen embrittlement. When a traditional manufacturing process is used, such as machining, the surface layer sustains local plastic deformation.

The processes that affect surface integrity can be conveniently broken up into three classes: traditional processes, non-traditional processes, and finishing treatments. Traditional processes are defined as processes where the tool contacts the workpiece surface; for example: grinding, turning, and machining. These processes will only damage the surface integrity if the improper parameters are used, such as dull tools, too high feed speeds, improper coolant or lubrication, or incorrect grinding wheel hardness. Nontraditional processes are defined as processes where the tool does not contact the workpiece; examples of this type of process include EDM, electrochemical machining, and chemical milling. These processes will produce different surface integrity depending on how the processes are controlled; for instance, they can leave a stress-free surface, a remelted surface, or excessive surface roughness. Finishing treatments are defined as processes that negate surface finishes imparted by traditional and non-traditional processes or improve the surface integrity. For example, compressive residual stress can be enhanced via peening or roller burnishing or the recast layer left by EDMing can be removed via chemical milling.

Finishing treatments can affect the workpiece surface in a wide variety of manners. Some clean and/or remove defects, such as scratches, pores, burrs, flash, or blemishes. Other processes improve or modify the surface appearance by improving smoothness, texture, or color. They can also improve corrosion resistance,wear resistance, and/or reduce friction. Coatings are another type of finishing treatment that may be used to plate an expensive or scarce material onto a less expensive base material.

Variables

Manufacturing processes have five main variables: the workpiece, the tool, the machine tool, the environment, and process variables. All of these variables can affect the surface integrity of the workpiece by producing:

- High temperatures involved in various machining processes
- Plastic deformation in the workpiece (residual stresses)
- Surface geometry (roughness, cracks, distortion)
- Chemical reactions, especially between the tool and the workpiece

4.9 Broaching Machines

- Broaching machines are relatively simple as they only have to move the broach in a linear motion at a predetermined speed and provide a means for handling the broach automatically. Most machines are hydraulic, but a few specialty machines are mechanically driven. The machines are distinguished by whether their motion is horizontal or vertical. The choice of machine is primarily dictated by the stroke required. Vertical broaching machines rarely have a stroke longer than 60 in (1.5 m).

- Vertical broaching machines can be designed for push broaching, pull-down broaching, pull-up broaching, or surface broaching. Push broaching machines are similar to an arbor press with a guided ram; typical capacities are 5 to 50 tons. The two ram pull-down machine is the most common type of broaching machine. This style machine has the rams under the table. Pull-up machines have the ram above the table; they usually have more than one ram. Most surface broaching is done on a vertical machine.
- Horizontal broaching machines are designed for pull broaching, surface broaching, continuous broaching, and rotary broaching. Pull style machines are basically vertical machines laid on the side with a longer stroke. Surface style machines hold the broach stationary while the workpieces are clamped into fixtures that are mounted on a conveyor system. Continuous style machines are similar to the surface style machines except adapted for internal broaching.
- Horizontal machines used to be much more common than vertical machines, however today they represent just 10% of all broaching machines purchased. Vertical machines are more popular because they take up less space.

4.10 Push Type Broaching Machine

Vertical internal push-down: Vertical push-down machines are often nothing more than general-purpose hydraulic presses with special fixtures. They are available with capacities of 2 to 25 tons, strokes up to 36" and speeds as high as 40 FPM. In some cases, universal machines have been designed which combine as many as three different broaching operations, such as push, pull, and surface, simply through the addition of special fixtures.

4.11 Pull Type Broaching Machine

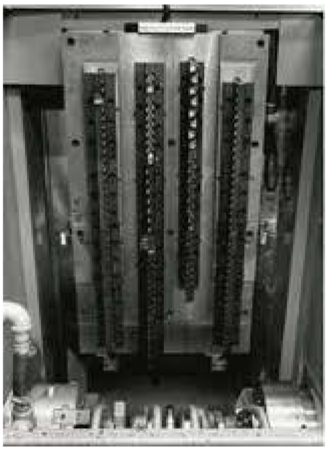
Vertical internal pull-up: The pull-up type, in which the workpiece is placed below the worktable, was the first to be introduced. Its principal use is in broaching round and irregular shaped holes. Pull-up machines are now furnished with pulling capacities of 6 to 50 tons, strokes up to 72", and broaching speeds of 30 FPM. Larger machines are available; some have electro-mechanical drives for greater broaching speed and higher productivity.

Vertical internal pull-down: The more sophisticated pull-down machines, in which the work is placed on top of the table, were developed later than the pull-up type. These pull-down machines are capable of holding internal shapes to closer tolerances by means of locating fixtures on top of the worktable. Machines come with pulling capacities of 2 to 75 tons, 30" to 110" strokes, and speeds of up to 80 FPM.

4.12 Surface broaches

The broaches used to remove material from an external surface are commonly known as surface broaches. Such broaches are passed over the workpiece surface to be cut, or the workpiece passes over the tool on horizontal, vertical or chain machines to produce flat or contoured surfaces.

While some surface broaches are of solid construction, most are of built-up design, with sections, inserts or indexable tool bits that are assembled end-to-end in a broach holder or sub holder. The holder fits on the machine slide and provides rigid alignment and support.



4.13 Continuous Chain Broaching

Continuous chain, or simply chain broaching refers to the type of machine that is used to broach a piece part.

Chain broaching is oriented towards high volume production, and is an extremely fast and efficient operation. However, because the fixtures used to hold the piece parts are mounted on chains that are driven by sprockets, it is difficult to hold extremely close tolerances. This process is suitable for high-volume, external cutting.

Continuous Chain Broaching Industries

- Biomedical
- Electronics
- Defense

A chain broaching machine resembles a very long tunnel, through which passes a series of holding fixtures, or cars. Piece parts are loaded, usually automatically, into the cars, which themselves are mounted on, and carried through the tunnel by a very large continuous chain. The broach tooling is mounted on the inside walls of the tunnel, and this tooling cuts the piece part as it passes through the tunnel. Contact us today to learn more.